Selecting nozzle arrangement of a chimney tower to reduce temperature and to increase entrainment mass flow

Sarjito, Waluyo Adi Siswanto, Agus Jamaldi, Yufeng Yao

***Abstract*— This research work aims to find the optimal arrangement of nozzles inside a chimney tower to reduce the temperature and to increase entrainment mass flow. An existing design of the nozzle arrangement has been used as a benchmark. Six alternative configuration models have been evaluated by using Computational Fluid Dynamics (CFD) analysis tool to quantify the effect of different configurations on the variations of temperature and entrainment mass flow. The parameters examined include temperature distribution inside the chimney, in conjunction with evaporation cooling system and the flow velocity. A precursor mesh influence study with five meshing types has been carried out, prior to the analysis of other nozzle arrangements, followed by main simulations. . The results of new nozzle configuration with 8 nozzles evenly distributed at the top surface layer and 3 nozzles at a height of 3.5 m from the base platform of the stack have shown the production of optimum capacity in decreasing temperature and increasing the rate of water vapor.**

***Index Terms*— Evaporative cooling, Chimney, Entrainment, Nozzle, CFD.**

I. Introduction

A multi-stage down-draft evaporative cool tower (DECT) is one key component for air conditioning of a building by employing fresh air blow passing through it. The use of this component is considered environmentally friendly as there is no need of electric power source. Basically the multi-stage model is further development of a predecessor single-stage model in order to improve its performance. In this respect, some design and investigations have been reported in various publications, including experiments [1, 2, 3] and as well as simulation computational fluid dynamics (CFD) approaches [4, 5, 6].

In order to improve the cooling effect, additional water spray from the prescribed nozzles into the chimney can be useful [7]. The water spray gives better cooling effect compared with the use of natural airflow only. This is because that the misting process or spraying water produced by nozzle configuration inside the chimney will directly contact with 'warm' air for convection and cooling.

Gant [8] reported simulation results of single spray by using CFD. The water spray, thermal energy transfer and momentum transport phenomena occurred between the spray and the ambient air were numerically simulated. The study investigated a computational domain of vertical cylinder diameter of 1 m and length of 1.5 m, using single nozzle vertically downward positioned at upper location of the cylinder domain. Meanwhile the nozzle geometry was a small cylinder with a diameter of 0.00625 m and a length of 0.05 m.

A comparison between CFD and experiment was previously performed [9], showing that simulation results were in good agreements with those from experiments. Tambur and Guetta [10] also reported their work using two commercially available nozzles of Bete PJ32 and TF6 for their experiments. Each nozzle was varied in orifice diameter to result in as many as 16 variation pressures. Their results indicated that the spray performance of PJ32 nozzle is better than TF6.

Another observation of water spray conducted by Pearlmutter et al. [11] showed that the highest temperature reduction in an experimental tower of a height 10 m occurred when the location of the spray at a height of 2 m from the ground. Above that location, the temperature reduction is not significantly changed.

Sarjito [13] has investigated the multi-stage tower using a number of nozzles, and configuration/position of nozzle to get optimum performance. Parameters investigated were mass flow rate, and uniformity velocity profile resulted by spray nozzle effect, etc. For different nozzle arrangements at the same pressure, the total mass flow remains the same, and was balanced by the number of nozzle used (i.e. mass conservation). The tower height was varied in a range of 3 - 4 meter. The commercial nozzle of TF6 was used at a working pressure of 3.33 bar, mass flow rate (air) of 0.096 kg/s and a spray velocity of 21.57 m/s, respectively.

Two basic arrangements of the nozzles were investigated; i.e. a configuration, in which a constant radius was maintained for the nozzle pitch circle while more nozzles were added, and a configuration, in which a constant spacing was maintained between all nozzles. It was found that the constant radius gave more temperature reduction and more induced mass flow rate, thus to produce this effect the number of nozzle should be added for configuration with constant distance. The use of nozzle numbers of 6 to 11 on an arrangement with constant spacing is found much more effective in reducing temperature and inducing mass flow rate. The best performance of cooling was observed for an arrangement with 9 nozzles and the most effective mass flow rate was found when using 10 nozzles.

Manuscript received August 20, 2018. This work was supported in part by the Universitas Muhammadiyah Surakarta under the publication incentive program.

Sarjito is with the Department of Mechanical Engineering, Universitas Muhammadiyah Surakarta, Indonesia. (e-mail: sarjito@ums.ac.id).

Waluyo Adi Siswanto is with the Department of Mechanical Engineering, Universitas Muhammadiyah Surakarta, Kartasura, Indonesia.
(e-mail: waluyo@ums.ac.id).

Agus Jamaldi is with the Department of Mechanical Engineering, Universitas Muhammadiyah Surakarta, Indonesia.
(e-mail: ajamaldi@yahoo.co.id).

Yufeng Yao is with the Faculty of Environmental and Technology University of the West of England, United Kingdom.
(e-mail: Yufeng.Yao@uwe.ac.uk)..

Further study by introducing one nozzle at the center gave more entrainment mass flow, therefore the use of 11 nozzles was finally selected.

Further research extending to multi-stage model was conducted by Sarjito and Marchant [14] to establish a baseline performance of cooling tower. The main parameter was the ratio of the secondary to the primary mass flow rates. The work presented the use of CFD to optimize the geometry of a multi-stage evaporative cooling device. In particular, the effects on the performance of varying the primary inlet to mixing stack area ratio. Both temperature and relative humidity (RH) contours at the cooled space and the velocity profiles at the outlet of the device were represented by real conditions such as the un-evaporated water and sensible cooling power. Both the simulation and calculation results have shown good agreement, compared with available test data.

The improvement of the performance of the multi-stage downdraught evaporative coolers was studied and simulated by employing CFD analysis then verified experimentally [15]. Those preliminary CFD work focused on establishing a correlation between environmental wind velocity and the down-draft quantity by comparing the result with the secondary experimental data. The detailed flow features that did not available from experiment was obtained and shown by numerical CFD study. The results have further extended the spraying model both of numerical and experimentally conducted by Gonzales-Tello et al. [16].

The research presented in this paper investigates alternative nozzle arrangement design for better temperature distribution and velocity profile in a chimney. It is also searching for a better level of RH. The research will be carried out computationally using computational fluids dynamics (CFD).

II. METHODOLOGY

A. Existing Nozzle Arrangement

An existing experimental chimney test rig with its nozzle arrangement was previously developed [13]. Two types of nozzle arrangement were available. In the first model, nozzles were set around a circle with a constant radius of 0.65 meters. The nozzles in second model was scattered with constant distance of 0.65 meters between the adjacent nozzles.

Figure 1 shows the nozzle arrangement consisting 11 nozzles. One nozzle is located at the center of chimney's vertical axis and the other ten nozzles are at a radius of 0.75 m with the distance of 0.65 m between two adjacent nozzles.

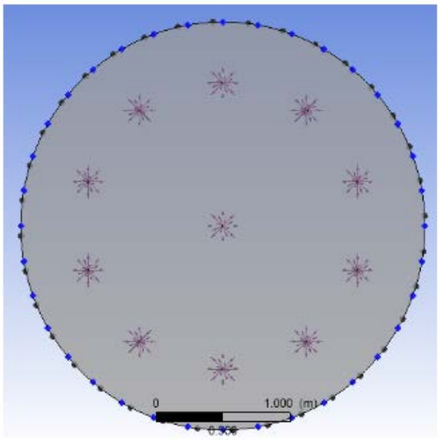


Fig. 1. Eleven nozzle arrangement.

The illustration of the chimney and the vertical locations of the nozzles are shown in Fig. 2.

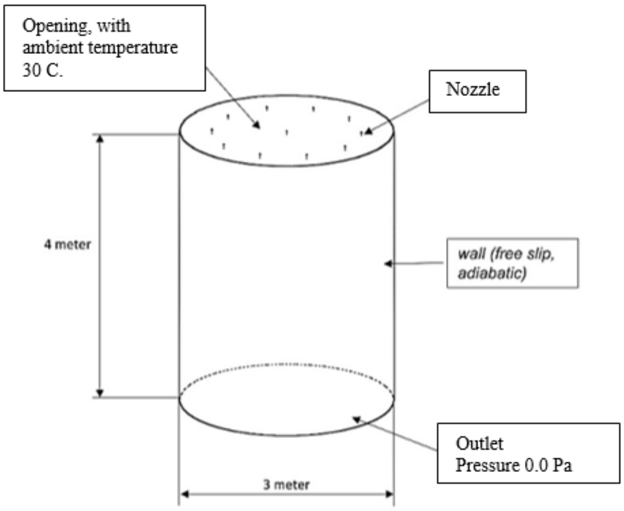


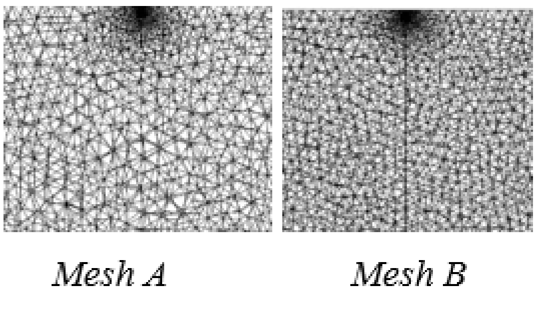
Fig. 2. Chimney and nozzle location.

B. Meshing Sensitivity

In computational analysis, element mesh size is very important to obtain accurate results. Five meshes design from coarse to fine were prepared.

Simulations were conducted for all meshing types and compared with the existing design, and the mesh producing smallest difference was then selected for further analysis.

The meshing models indicated as meshes A, B, C, D, and E are illustrated in Fig. 3.



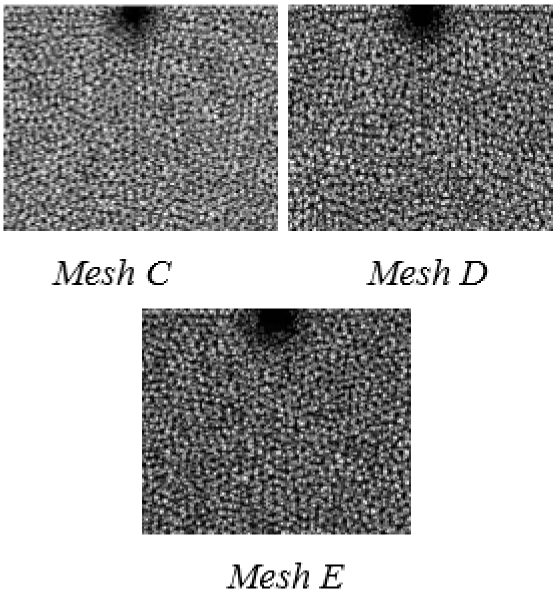


Fig. 3. Chimney and nozzle location.

The comparison information of meshing types including the number of nodes and elements are depicted in Table 1.

TABLE I: MESHING PROPERTIES COMPARISON

Tipe	Mesh A	Mesh B	Mesh C	Mesh D	Mesh E
Min. size	0.006	0.004	0.0029	0.0014	0.00085
Max. size	0.11	0.08	0.05	0.05	0.045
Nodes	90639	204853	348068	353747	485360
Elements	515220	1179441	2016345	2049022	2820115

C. *Study on Nozzle Configuration*

In order to improve the performance of the nozzles, several configurations were tried to replace the eleven nozzles arrangement located on the same layer. The alternative configurations will have the same number of nozzles, but at different locations.

Three configurations have been proposed and tested as:

a. 4 nozzles on the top layer and 7 nozzles at the lower layer
Seven nozzles were tried in 6 alternative heights from the bottom: 1 m, 1.5 m, 2 m, 2.5 m, 3 m and 3.5 m. The illustration of these models is illustrated in Fig. 4.

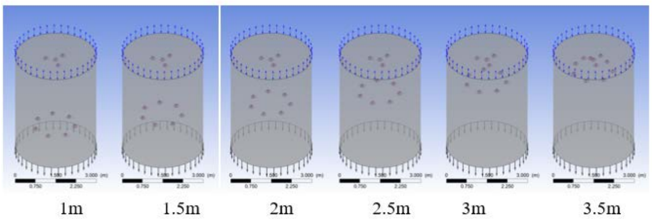


Fig. 4. Nozzle configuration 4-7

b. 6 nozzles on the top layer and 5 nozzles at the lower layer
Five nozzles were tried in 5 alternative heights from the bottom: 1 m, 1.5 m, 2 m, 2.5 m, 3 m and 3.5 m. The illustration of these models is illustrated in Fig. 5.

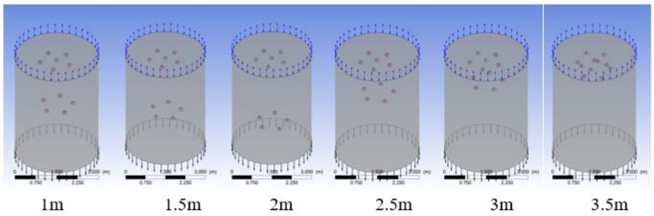


Fig. 5. Nozzle configuration 5-6

c. 8 nozzles on the top layer and 3 nozzles at the lower layer
Three nozzles were tried in 6 alternative heights from the bottom: 1 m, 1.5 m, 2 m, 2.5 m, 3 m and 3.5 m. The illustration of these models is illustrated in Fig. 6.

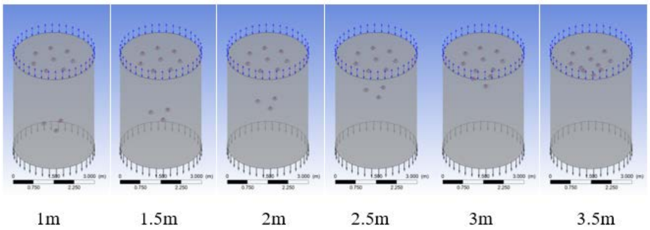


Fig. 6. Nozzle configuration 8-3

d. 11 nozzles in a helical pattern
Eleven nozzles were arranged in a helical pattern as seen in Fig. 7. The nozzles are similar to a rotating helical configuration.

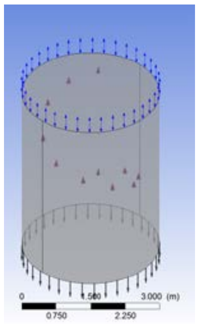


Fig. 7. Nozzle helical configuration.

III. RESULTS AND DISCUSSION

A. *Meshing Selection*

Simulations using five meshes were performed and results compared against the experimental measurement previously conducted, taken from reference [13]. The comparison of simulation results from meshes A, B, C, D and E and the experiment is shown in Table 2 and plotted in Fig. 8.

TABLE II: MESHING COMPARISON WITH EXPERIMENT

Measurement level from top of tower (m)	Temperature (K)					
	Sarjito [13]	*Mesh A*	*Mesh B*	*Mesh C*	*Mesh D*	*Mesh E*
0	300.93	299.343	300.181	300.699	300.888	301.113
0.95	301.461	300.868	300.938	301.695	301.749	301.766
1.95	302.404	301.557	301.613	302.391	302.361	302.402
2.95	303.054	302.256	302.294	302.856	302.847	302.868
3.95	303.15	303.143	303.145	303.146	303.147	303.148

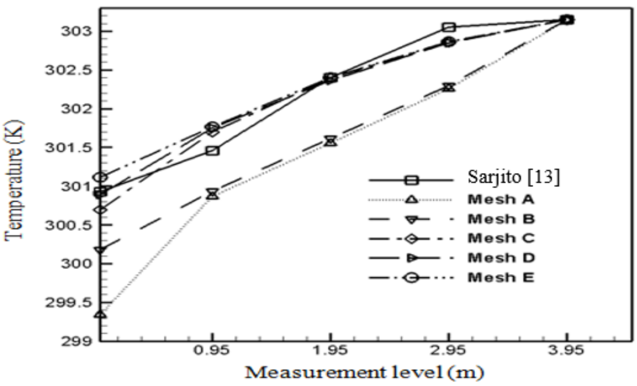


Fig. 8. Temperatures on various meshing.

The results of Table II and Fig. 8 indicate that the simulations using meshes A, B, C, D and E have shown patterns comparable to previous study [13]. However mesh C produces the closest data compared with those from experimental data. It is therefore decided to use mesh C for further analysis.

B. Nozzle configurations 4-7

The position of seven nozzles in lower layer at height levels were 1 m, 1.5 m, 2 m, 2.5 m, 3 m and 3.5 m, represented by Zmax-Z1, Zmax-Z2, Zmax-Z3, Zmax-Z4, Zmax-Z5 and Zmax-Z6 respectively. The temperature comparison results are depicted in Fig. 9.

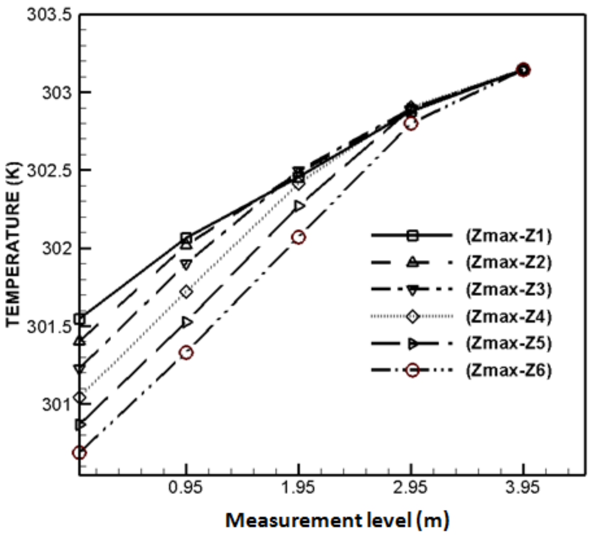


Fig. 9. Temperatures on 4-7 nozzle configuration.

Figure 9 shows that temperature reduction can be achieved at best at a position Zmax – Z6, where 7 nozzles are at a height of 3.5 m from the ground. At this position, the distance between the upper and the lower nozzles is 0.5 m. The maximum temperature reduction is 2.454 ^{o}C or about 8.18% reduction.

C. Nozzle configurations 6-5

In this configuration, 5 nozzles were positioned at 1 m, 1.5 m, 2 m, 2.5 m, 3 m and 3.5 m and indicated as Zmax-Z1, Zmax-Z2, Zmax-Z3, Zmax-Z4, Zmax-Z5 and Zmax-Z6 respectively. The simulation results are shown in Fig. 10.

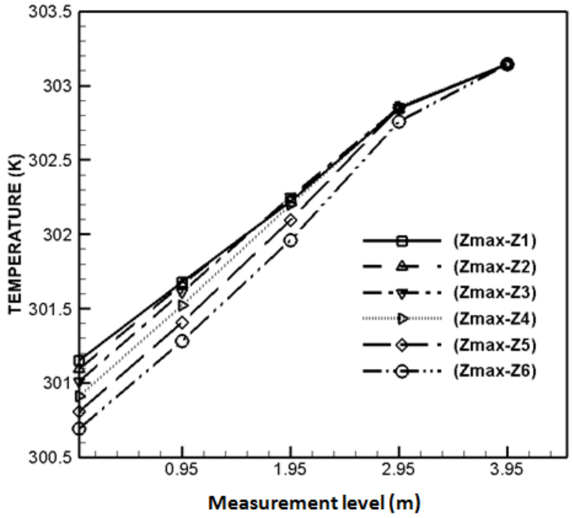


Fig. 10. Temperatures on 6-5 nozzle configuration.

By considering the simulation results shown in Fig. 10, it is found that the minimum temperature reduction occurred when the 5 nozzles at the level of 3.5 m from the ground (i.e. Zmax–Z6). In this configuration the temperature reduction can be achieved as much as 2.45 ^{o}C or decreased by 8.17%

D. Nozzle configuration 8-3

Three nozzles were tried at the level of 1 m, 1.5 m, 2 m, 2.5 m, 3 m and 3.5 m from the ground, represented by Zmax-Z1, Zmax-Z2, Zmax-Z3, Zmax-Z4, Zmax-Z5 and Zmax-Z6 respectively. The simulation results are shown in Fig. 11.

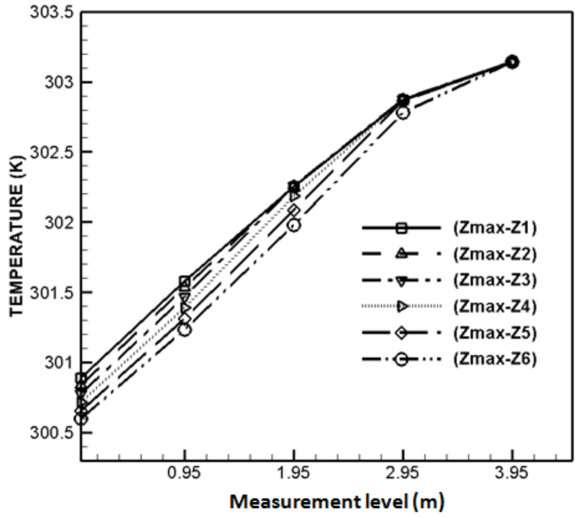


Fig. 11. Temperatures on 8-3 nozzle configuration.

The results from Fig. 11 show that when the three nozzles are at a height of 3.5 m from the ground, the temperature can be reduced by 2.55 ^{o}C or by 8.17% reduction. This is similar to that achieved from 4-7 configurations.

E. Nozzle configuration 11 Helical Pattern

Eleven nozzles were configured helically using a constant space from the top layer to the lower layer 0.5 from the ground. There was no nozzle in the center. The temperature distribution in different levels can be seen in Table III.

TABLE III: TEMPERATURE DISTRIBUTION 11 HELICAL NOZZLES

Level from top (m)	Temperature (K)
0	300.21
0.95	301.58
1.95	302.50
2.95	302.98
3.95	303.15

It can be seen from Table III that helically nozzle configuration resulting the highest temperature of 303.15 K and the lowest temperature of 300.21 K. The temperature reduction can achieve as much as 2.94 ^{o}C or about 9.8% reduction.

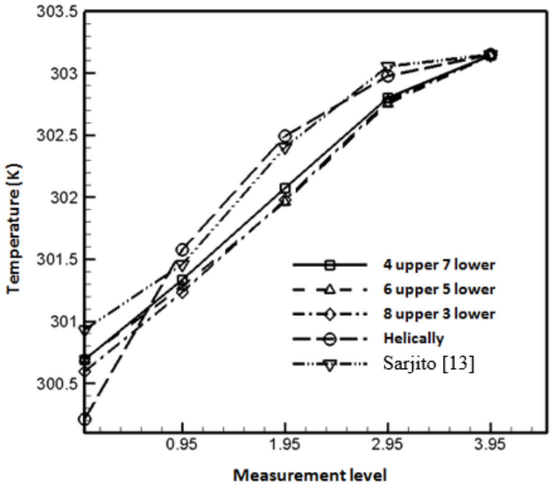


Fig. 12. Comparison of all configurations.

F. Selection Nozzle configuration

The comparison of all configurations are summarized and given in Fig. 12. The comparison chart shows that nozzle configuration resulting in the lowest temperature reduction was 8 nozzles on the top layer and 3 nozzles at a lower position (Zmax–Z6). The temperature reduction at the lower plane is 2.55 ^{o}C.

G. Effect of nozzle configuration to fluid entrainment

Further study in conjunction with the nozzle configuration to temperature was the entrainment mass flow rate of air inside the tower. The air velocity distribution for all configuration investigated is shown in Fig. 13.

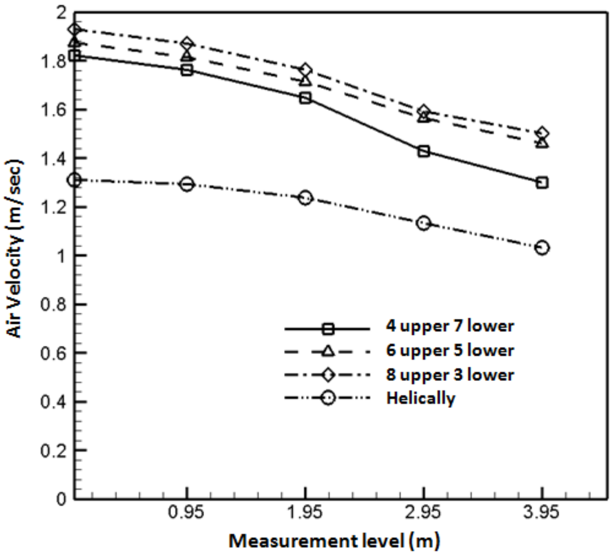


Fig. 13. Comparison of air velocity and nozzle configuration.

Based on Fig. 13, it is evident that air velocity inside the tower was increasing along with additional nozzle on the top of tower. The more nozzles used the higher mass flow rate, as the velocity and quantity of the mass flow rate entrained will be increased.

IV. Conclusion

This work is to search for the optimal configurations of array nozzle spray and its effect particularly on the temperature reduction in cooling tower by using computational fluid dynamics simulation tool.

The simulation results show that the nozzle configuration resulting in the lowest temperature reduction was 8 nozzles on the top and 3 nozzles at a lower position (Zmax – Z6). This mode gives maximum temperature reduction of 2.55 ^{o}C at the lower plane, compared to other configurations studied.

Further study proved that air velocity inside the tower was increasing along with an additional nozzle on the top of tower. The more nozzles used the more mass flow rate increase.

Acknowledgment

Authors would like to thank to Universitas Muhammadiyah Surakarta (UMS) for providing research funding and supporting international publications (do you have grant number? If so, pls add here).

References

[1] E. Erell, D. Pearlmutter. Y. Etzion, "A multi-stage down-draft evaporative cool tower for semi-enclosed spaces: Aerodynamic performance", *Solar Energy*, Vol. 82, Issue 5, pp. 420-429, 2008.

[2] D. Pearlmutter, E. Erell, Y. Etzion, "A multi-stage down-draft evaporative cool tower for semi-enclosed spaces: Experiments with a water spraying system", *Solar Energy*, Vol. 82, Issue 5, pp. 430-440, 2008.

[3] A. R. Dehghani-sanij, M. Soltani, K. Raahemifar, "A new design of wind tower for passive ventilation in buildings to reduce energy consumption in windy regions", *Renewable and Sustainable Energy Reviews*, Volume 42, pp.182-195, February 2015.

[4] D. Kang, R. K. Strand, "Modeling of simultaneous heat and mass transfer within passive down-draft evaporative cooling (PDEC) towers with spray in FLUENT", *Energy and Buildings*, Vol. 62, pp. 196-209, July 2013.

[5] D. Kang, R. K. Strand, "Significance of parameters affecting the performance of a passive down-draft evaporative cooling (PDEC)

tower with a spray system", *Applied Energy*, Vol. 178, pp. 269-280, 15 September 2016.
[6] P. Srivastava, Y. Khan, M. Bhandari, J. Mathur, R. Pratap, "Calibrated simulation analysis for integration of evaporative cooling and radiant cooling system for different Indian climatic zones", *Journal of Building Engineering*, Vol. 19, pp. 561-572, September 2018.
[7] Y. Sun, Z. Guan, H. Gurgenci, K. Hooman, X. Li, "Investigations on the influence of nozzle arrangement on the pre-cooling effect for the natural draft dry cooling tower", *Applied Thermal Engineering*, Vol. 130, pp. 979-996, 5 February 2018.
[8] S.E. Gant, "CFD Modelling of Water Spray Barriers". Harpur Hill, Buxton Derbyshire, SK17 9JN. UK, 2006.
[9] M. St-Georges, J. M. Buchlin, "Detailed single spray experimental measurements and one-dimensional modelling". *International Journal of Multiphase flow*, Vol.20, Issue 6, pp. 979-992, 1994.
[10] Y. Tambur, S. Gueta, "Optimizing the design and operation of the sprayers in the tower". Faculty of Aerospace engineering, Tachnion-Israel Institute of Technology, Appendix B., Israel, 2006.
[11] D.Pearlmutter, E.Erell, Y.Etzion, I.A.Meir, H.Di, "Refining the use of evaporation in an experimental down-draft cool tower", *Energy and Buildings*, Vol. 23, Issue 3, pp. 191-197, March 1996.
[12] M.N. Bahadori, An improved design of wind towers for natural ventilation and passive cooling. Solar Energy, (35), pp. 119–129.
[13] Sarjito. "An investigation of the design and performance of a multi-stage downdraught evaporative cooler". PhD Thesis, Faculty of Science Engineering and Computing, Kingston University, London.
[14] Sarjito, D. Marchant, "A study of a multi-stage downdraught evaporative cooling device using CFD to assess the effect of the primary inlet to mixing stack area ratio: Preliminary work to establish the baseline performance of the device", *Applied Mechanics and Materials*, Vol 465-466. pp 248-254, 2013.
[15] Sarjito, "An investigation of the scope for improvement of the performance of multi-stage downdraught evaporative coolers using CFD", *Applied Mechanics and Materials*, Vol 315, pp 835-840, 2013.
[16] P. Gonzales-Tello, F. Camaco,, J.M Vicaria,. "A modified Nukiyama-Tanasawa distribution function and Rosin-Rammler model for the particle-size-distribution analysis", *Powder Technology*, Vol. 186, Issue 3, pp. 278-281, September 2008.